Nonlinear Analysis with SOL 106

A Seminar for Femap and NX Nastran Users

Adrian Jensen, PE, Senior Staff Mechanical Engineer
V. Prathyush BheemReddy, PhD, Senior Staff Mechanical Engineer
George Laird, PhD, PE, Principal Mechanical Engineer

FINITE ELEMENT ANALYSIS
Predictive Engineering
LS-DYNA Sales, Support & Consulting
www.PredictiveEngineering.com

AppliedCAx
Siemens PLM Software Sales & Support
CAD I CAM I CAE I Teamcenter
www.AppliedCAx.com

What this white paper covers:

This paper is intended for NX Nastran users interested in exploring nonlinear solutions for their structural analysis. This note is intended to accompany a technical seminar and will provide you a starting background on nonlinear analysis.

The following topics are covered:

- Geometric Nonlinearity
  - Contact
  - Buckling
- Material Nonlinearity
  - Plasticity
  - Hyperelastic Materials
- Multi-Step Analysis
  - Contact with Bolt Preload
  - Vibration of Stiffened Structures
  - Residual Stresses from Plastic Deformation
- Gap Elements
  - Compression Only (Contact)
  - Tension Only (Cables)

TABLE OF CONTENTS

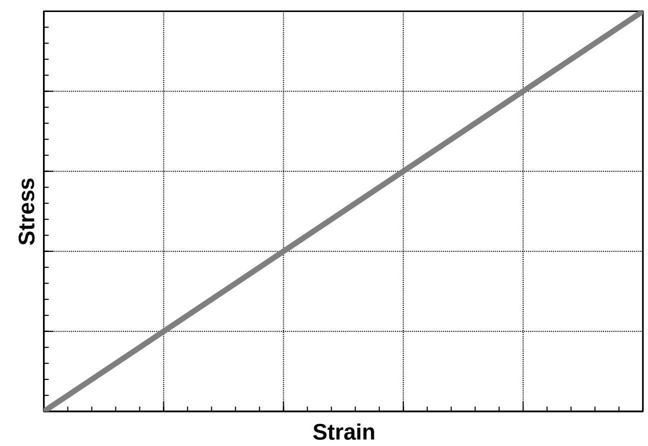
1. INTRODUCTION

1.1 Linear Elastic FEA

Before diving into nonlinear analysis, it is wise to get a good handle on linear analysis and understand its limitations.

Some important notes about linear analysis:

- Stresses can be scaled as a linear function of the loads
- Displacements can be scaled as a ratio of elastic moduli
- You can take advantage of superposition
- The structure is elastic (it never yields)



Stress is independent of your material choice.

Application: If the load is a force/pressure, then the resulting stress is just a function of geometry and not the material (i.e., homogeneous materials).

$$\sigma = \frac{Force}{Area}$$

Static means no acceleration

Application: Your structure must be constrained in all six DOFs (translation and rotation). If it is not constrained correctly – it cannot be solved.

$$\sum F = 0$$

2. INTRODUCTION TO NONLINEAR ANALYSIS TYPES

2.1 Geometric Nonlinear

How is "large displacement" defined? What happens to the FEA model when we perform a large displacement analysis? We will discuss what the solver is really doing with the stiffness matrix and how it will affect your results.

Figure 1 provides two sets of results for a pressurized plate with pinned connections at both ends. For context, the plate is 10" long, 1" wide and 1/16" thick with a pressure load of 10 psi.

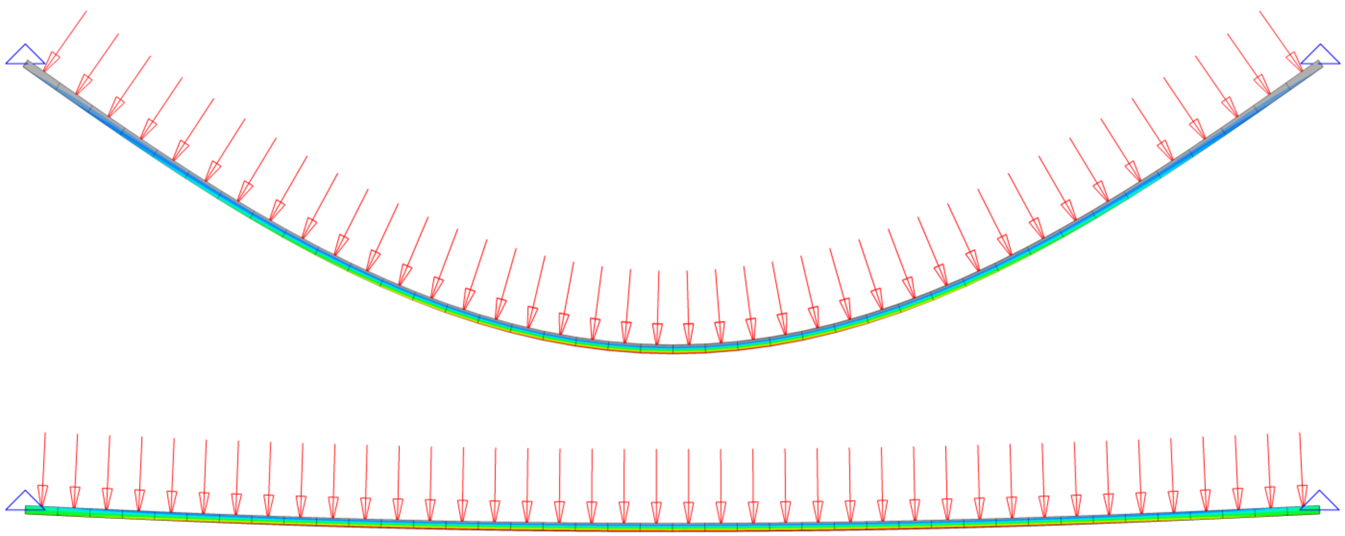


Figure 1: Linear and Geometric Nonlinear results for a pressurized plate

The linear analysis calculates a deflection of 2.2" and a maximum stress of 190,000 psi. The geometric nonlinear analysis (no plasticity) calculates a deflection of 0.1" and a maximum stress of 24,000 psi. There is a disparity in these results. What is happening in reality? How would these results differ if the plate was simply supported? What if the load was consolidated at the center?

From the Basic Nonlinear Analysis User's Guide:

"Geometric nonlinear effects are prominent in two different aspects: geometric stiffening due to initial displacements and stresses, and follower forces due to a change in loads as a function of displacements. These effects are included, but the large deformation effect resulting in large strains is implemented only for hyperelastic materials."

In Figure 2, a free body diagram (FBD) is displayed for a section-cut at the center of the plate. The FBD shows that linear analysis calculates pure bending while the nonlinear analysis develops tension in the plate.

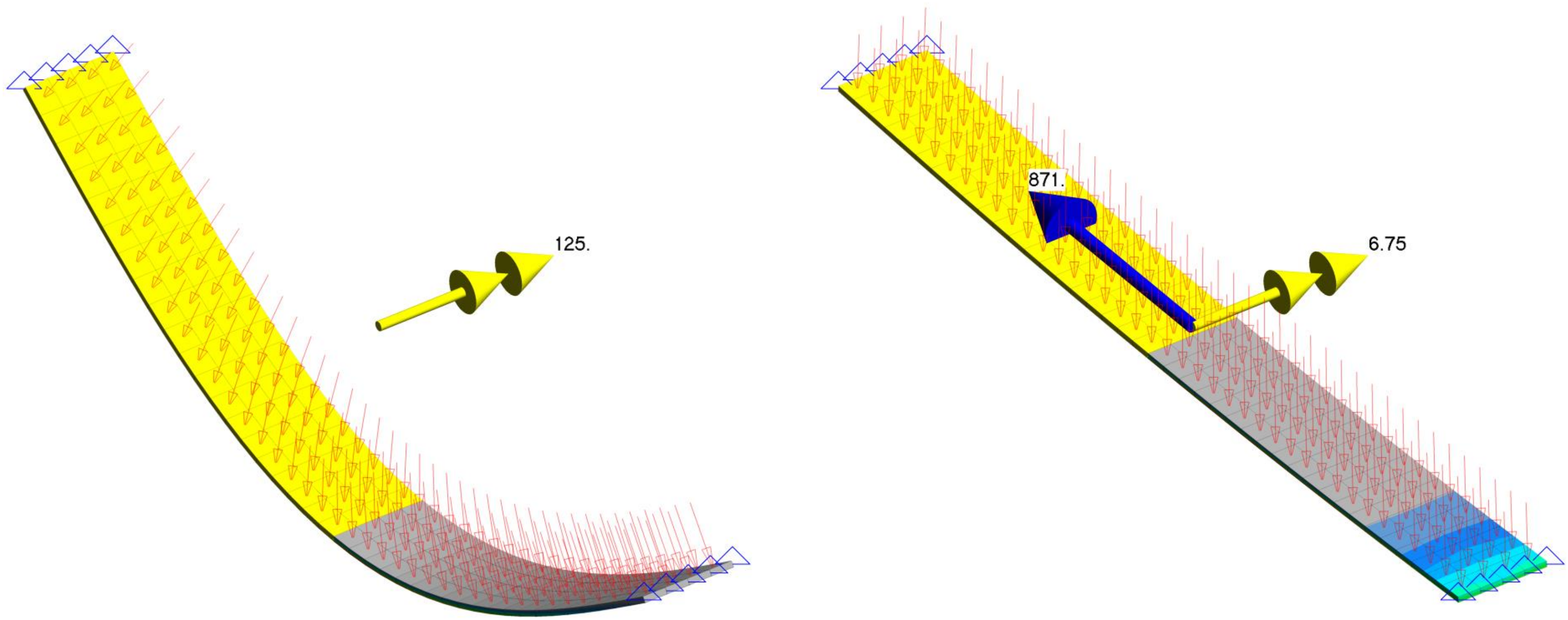


Figure 2: Linear and nonlinear results with a FBD at the center of the plate

2.1.1 NX Nastran Linear Contact: The Most Widely Used Nonlinear Analysis

Incorporating contact between components is a great first step into the nonlinear world. Although it is used with the linear static Nastran solver (SOL 101) it does use an iterative approach to determine the load path through the connections. In our first example, we will analyze a pin and clevis with contact as shown in Figure 3.

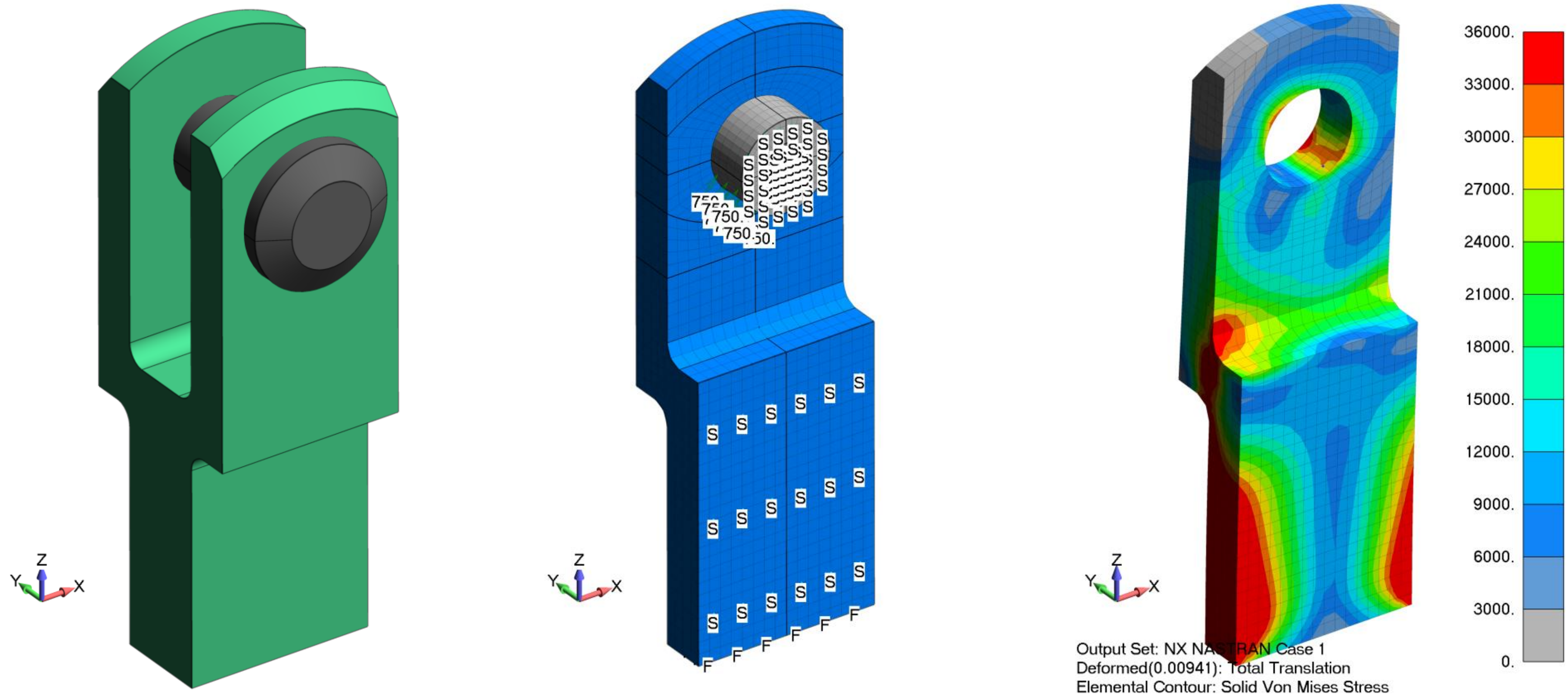
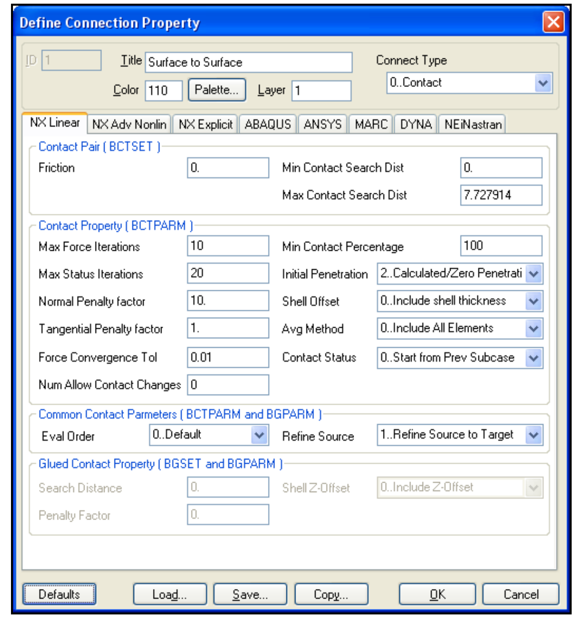
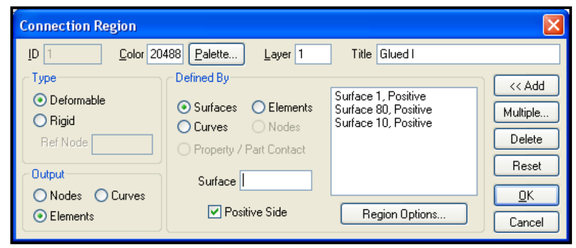


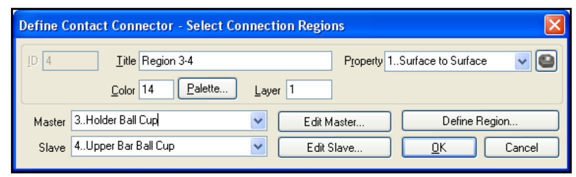
Figure 3: Pin/clevis contact problem as solved in SOL 101 with linear contact

NX Nastran Linear Contact for Solution Sequence 101:

- It is almost necessary to read the manual…. (NX Nastran User's Guide, Chapter 19).
- The NX Nastran linear contact has a legacy directly from SDRC-Ideas. It is a well-proven technology and they are still improving it.
- Be careful of the defaults for Min and Max Contact Search Dist. Actual contact elements are created within the solver based on the regions that you create in Femap and then the distances specified in these entries. One can create virtual contact elements where one least suspects it….
- Num Allow Contact Changes default of 0.0 is often too rigorous. One suggestion from an experienced user is that it should be 1% of the number of contact elements formed within the solver (see your F06 file – it will be listed in the first contact iteration dialog).
- Setting up contact regions can be done automatically but it may be more effective in the long run to carefully select your contact regions via the Connection Region dialog boxes.
- Think of the total process as 1.Connection Property; 2.Region; and 3.Connector.







2.1.2 Buckling

Buckling occurs when the structure becomes unstable and can no longer resist the applied load. Typically this type of behavior occurs in structures under high compressive load: classic examples are submersibles and load bearing towers (see Figure 4). Linear buckling theory takes into effect the very real physical mechanism of stress stiffening due to tensile loads or by reducing the stiffness due to compressive axial loads. This differential stiffness is a function of the geometry and applied loads.

$$[K_a] + \{P_{critical}\}[K_{differential}] = 0$$

However, the key assumption for linear buckling is that deflections are small up to the point of buckling and that the stress levels are below the yield point of the material. What if this is not the case? Nonlinear analysis can expand the capabilities of buckling analysis and provide verification on your linear buckling calculations.

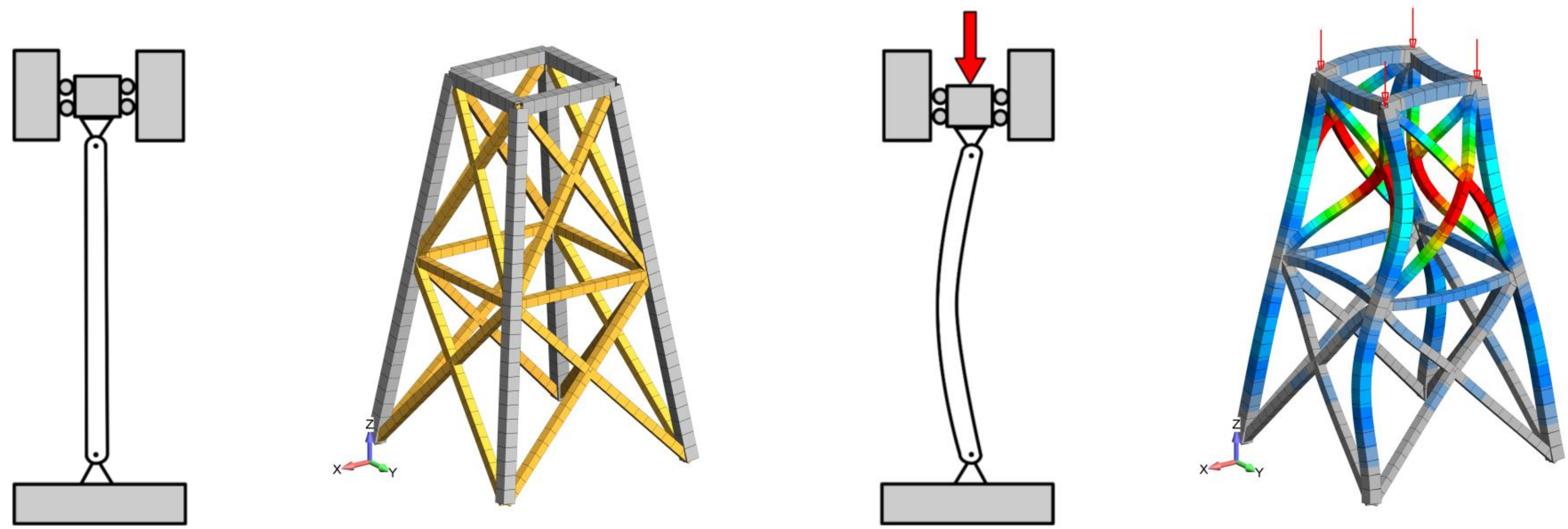


Figure 4: Examples of Euler column and tower buckling

In linear buckling, a differential stiffness matrix is used to determine the structure's potential response to a certain applied load. In a geometric nonlinear analysis, the stiffness matrix includes these differential stiffness terms and is updated at each load step. As the load is applied, the structure will deform. This deformed shape is numerically incorporated into the stiffness matrix.

In this example, we will explore the buckling of a simply supported column. The analysis setup is simple (see Figure 5) but the trick comes in the boundary conditions.

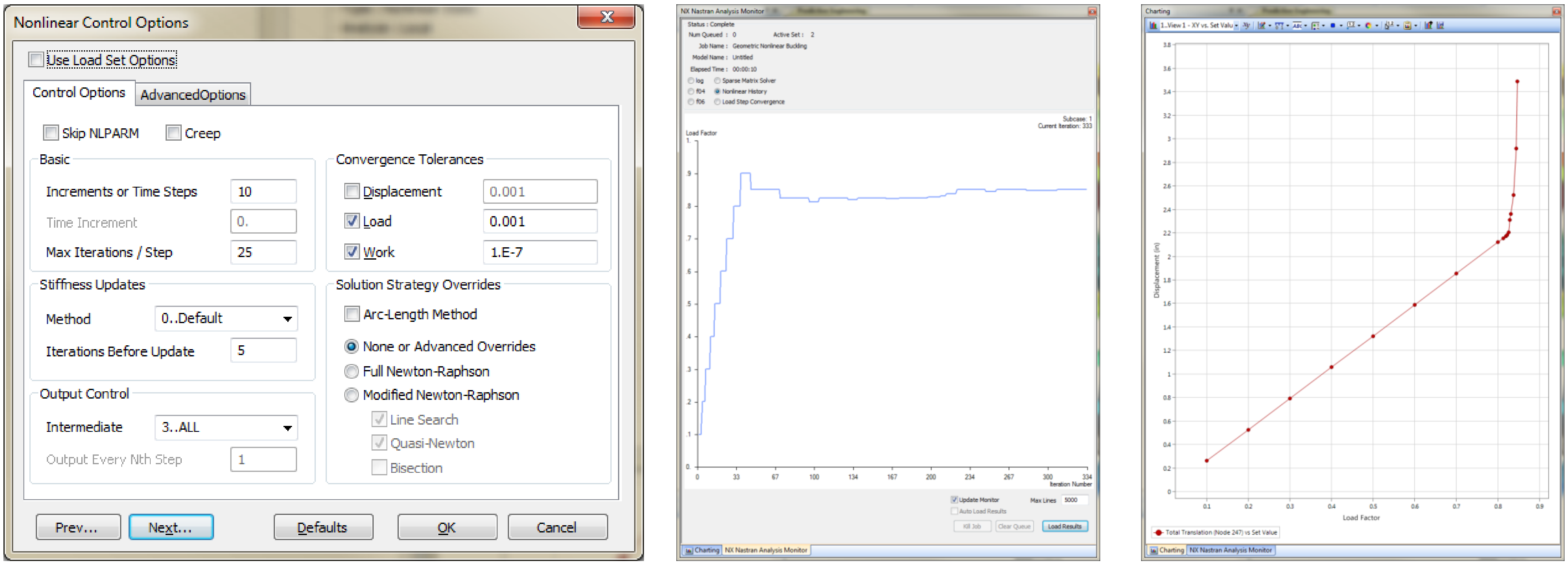


Figure 5: The key is to request "3..ALL" intermediate output in the Nonlinear Control Options

From the Basic Nonlinear Analysis User's Guide:

"It is recommended to check all possible buckling modes with the linear buckling solution sequence SOL 105 before making a run with SOL 106. You have to introduce initial imperfections (loads or enforced displacements) to keep the structure in the intended buckling mode."

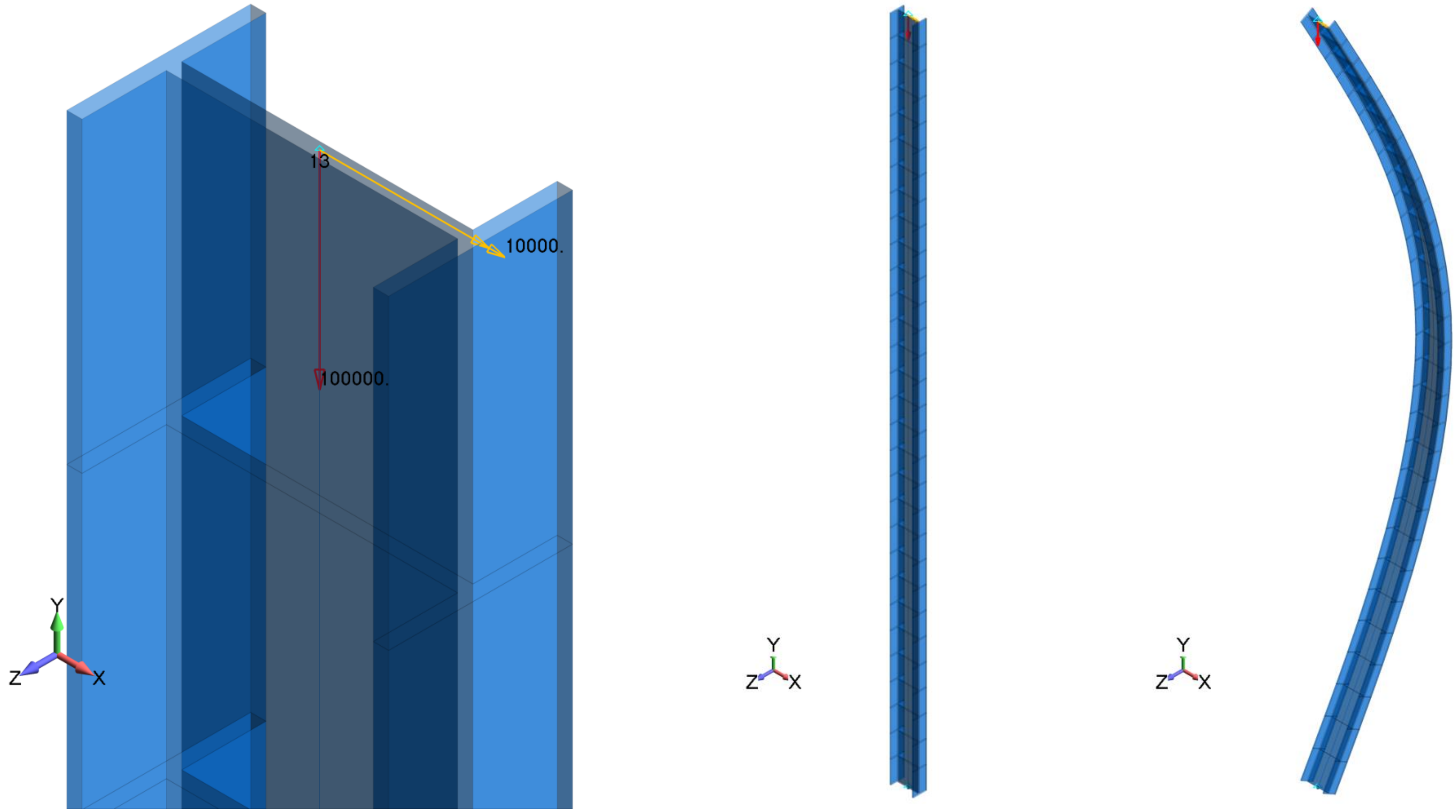


Figure 6: A bending moment at the top of the column was necessary to instigate buckling

2.2 Material Nonlinearity

"The stresses and strains of the small strain elements refer to the undeformed area and length, respectively."

Figure 7 shows how the solver tracks the translation and rotation of elements during a geometric nonlinear (large displacement) analysis. This ties into a material nonlinear analysis; since the solver uses the undeformed area of the element, the analyst must use engineering stress and strain when working with small strain elements.

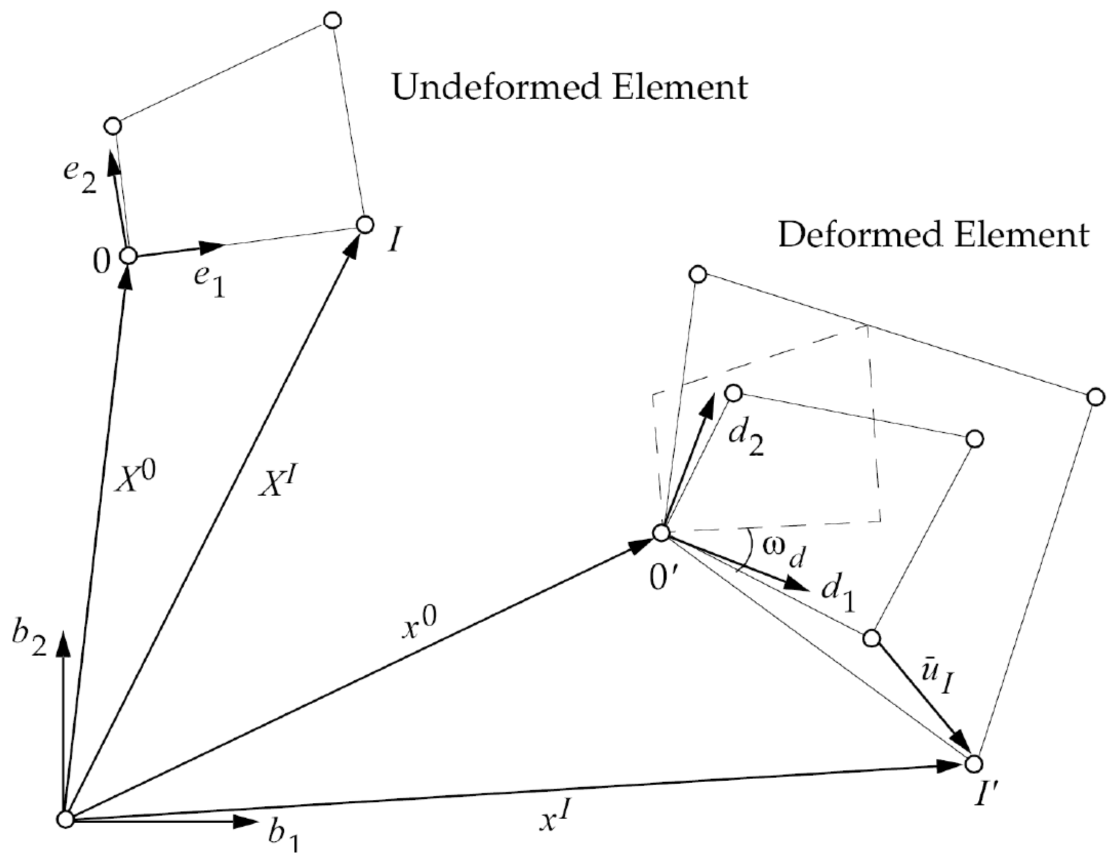


Figure 7: Co-rotational concept for small strain elements in geometric nonlinear analysis

2.2.1 Metal Plasticity (Bi-Linear or Engineering Stress-Strain Curve)

The most common application of material nonlinearity for analysts is the incorporation of plasticity into their material models. Fortunately, this is a simple idealization if one knows the yield stress, ultimate tensile strength and strain at failure. In this example, we will explore the bi-linear and the plastic nonlinear material methods with a simple four-point bending test as shown in Figure 8.

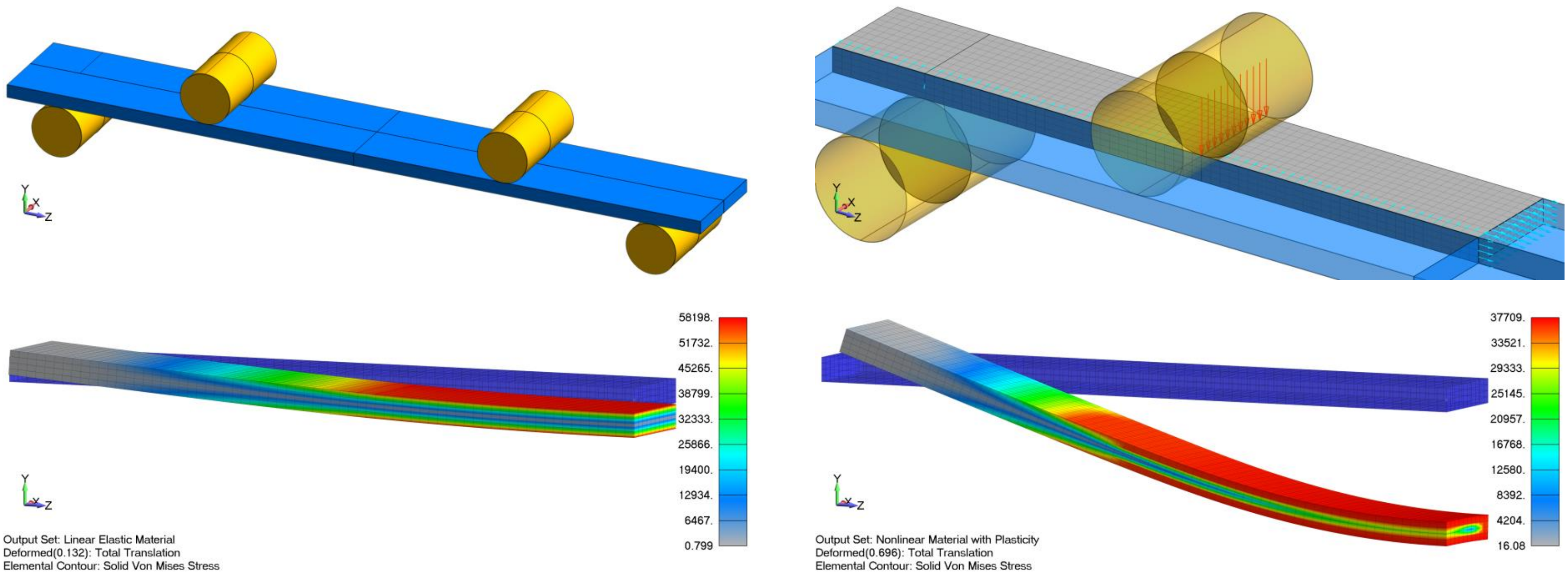


Figure 8: Four-point bending test with material plasticity

The model used in this example is ready to perform a linear analysis; all we need to do is create the nonlinear material model. We will assume the material is A36 steel with a yield stress of 36,000 psi, an ultimate tensile strength of 60,000 psi and 20% strain at failure.

2.2.2 Elastomers or Rubbers (Hyperelasticity)

"Large element net deformations should be avoided. In areas of the structure where large total deformations are expected, the mesh must be fine enough to keep the element net deformations small. The element net rotation should not exceed 20 degrees and the element should not be stretched by more than 10%. If stretches exceed 20%, it is recommended to use Hyperelastic elements if applicable."

If you have an engineering problem that requires the use of Hyperelastic materials, feel free to contact us!

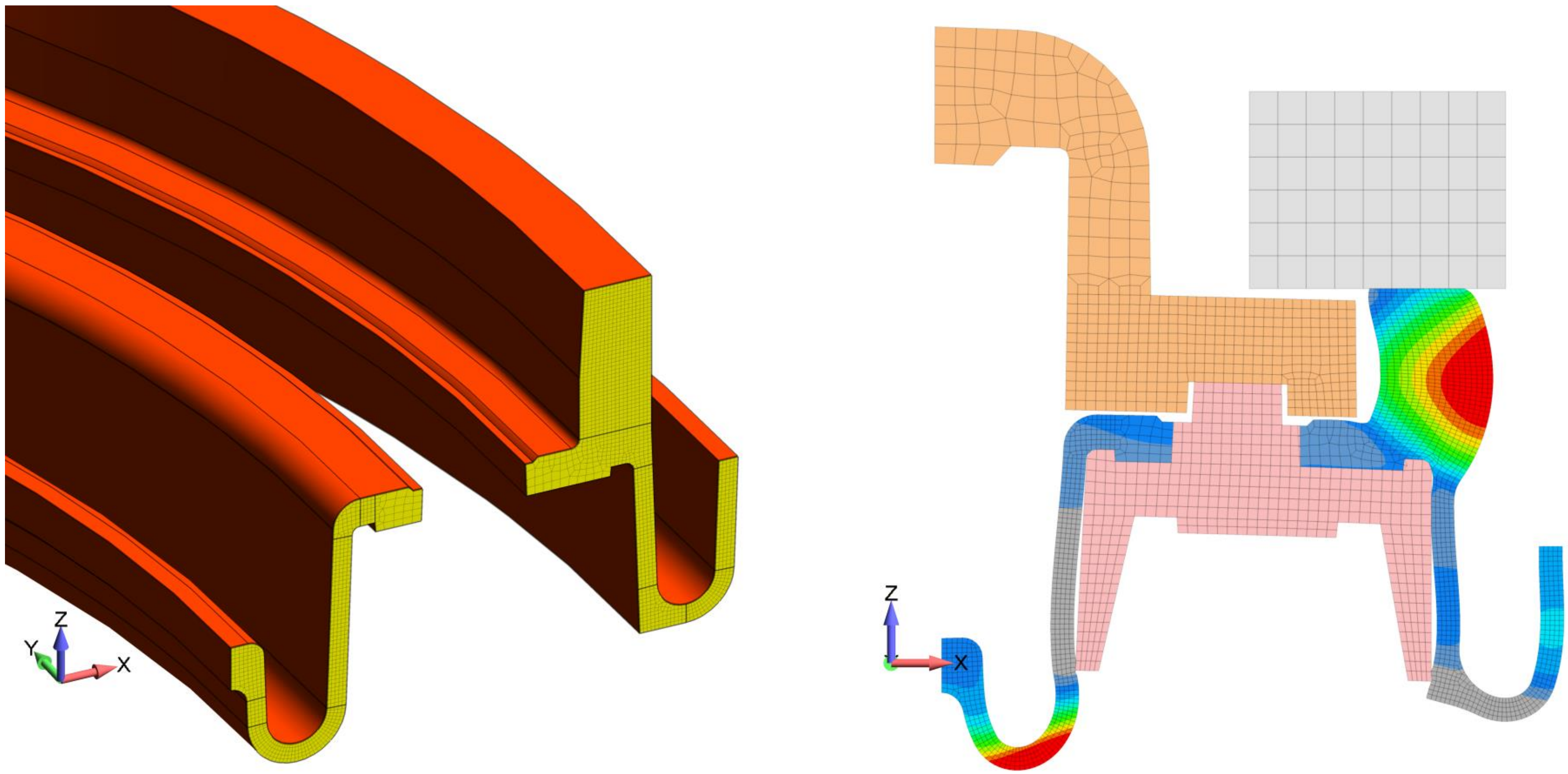


Figure 9: Example Predictive Engineering consulting model

3. NONLINEAR ANALYSIS EXAMPLES

3.1.1 CONTACT WITH BOLT PRELOAD (NO PLASTICITY)

With this example, we are back to the linear static solver (SOL 101) but this time, we are incorporating some more nonlinear behaviors. We have already explored linear contact but how do we incorporate bolt preload into the model? How will bolt preload stiffen the structure and affect the stresses? We will be working with the bearing pedestal seen in Figure 10.

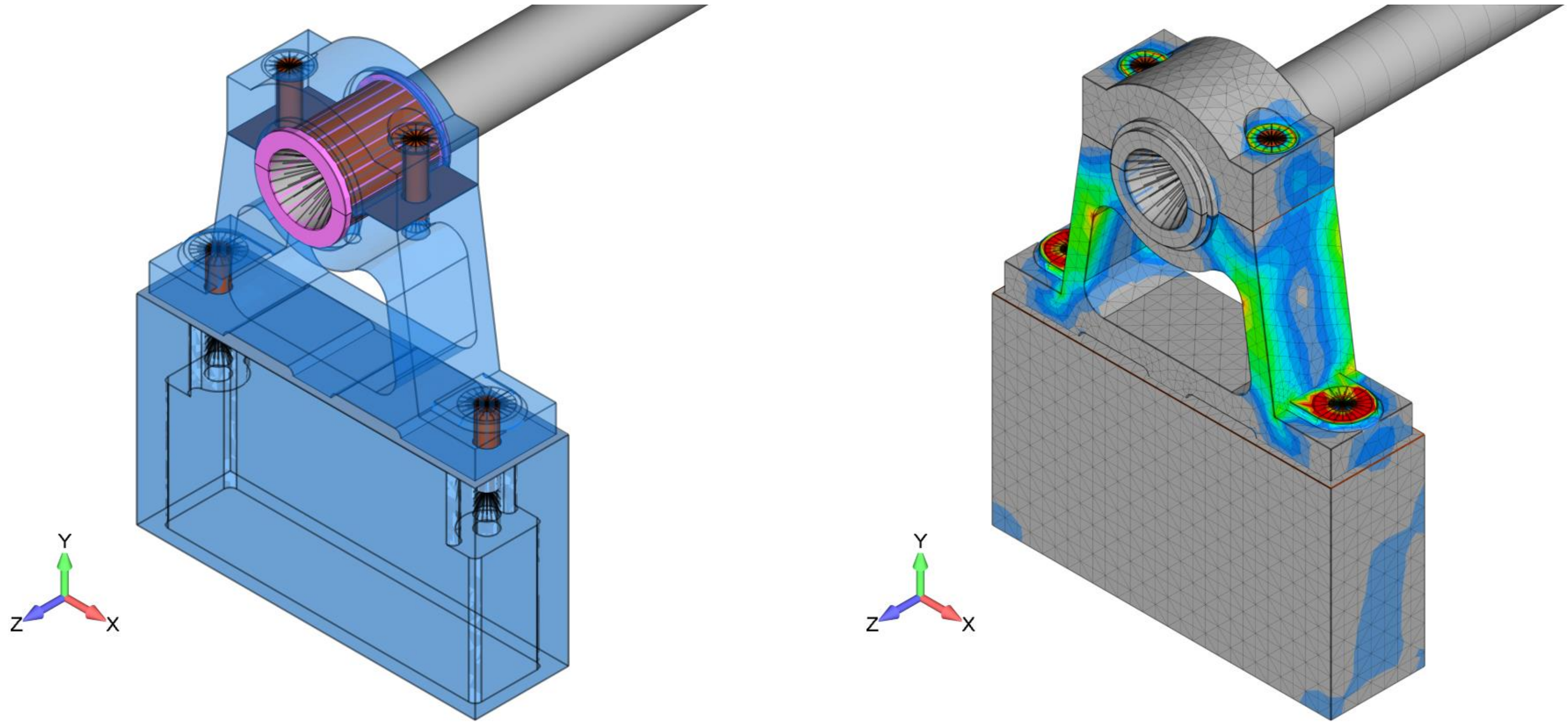


Figure 10: A simple shaft supported by a pillow block bearing at each end

3.1.1 Multi-Step Analysis (Eigenvalue)

Bolt preload can also be used to improve the stiffness of the model for vibration analysis. Although the solver isn't enforcing contact during the normal modes analysis (you can't simulate a "clapping" event), it is incorporating the updated stiffness matrix. Figure 11 shows how including bolt preload can have a significant effect on results.

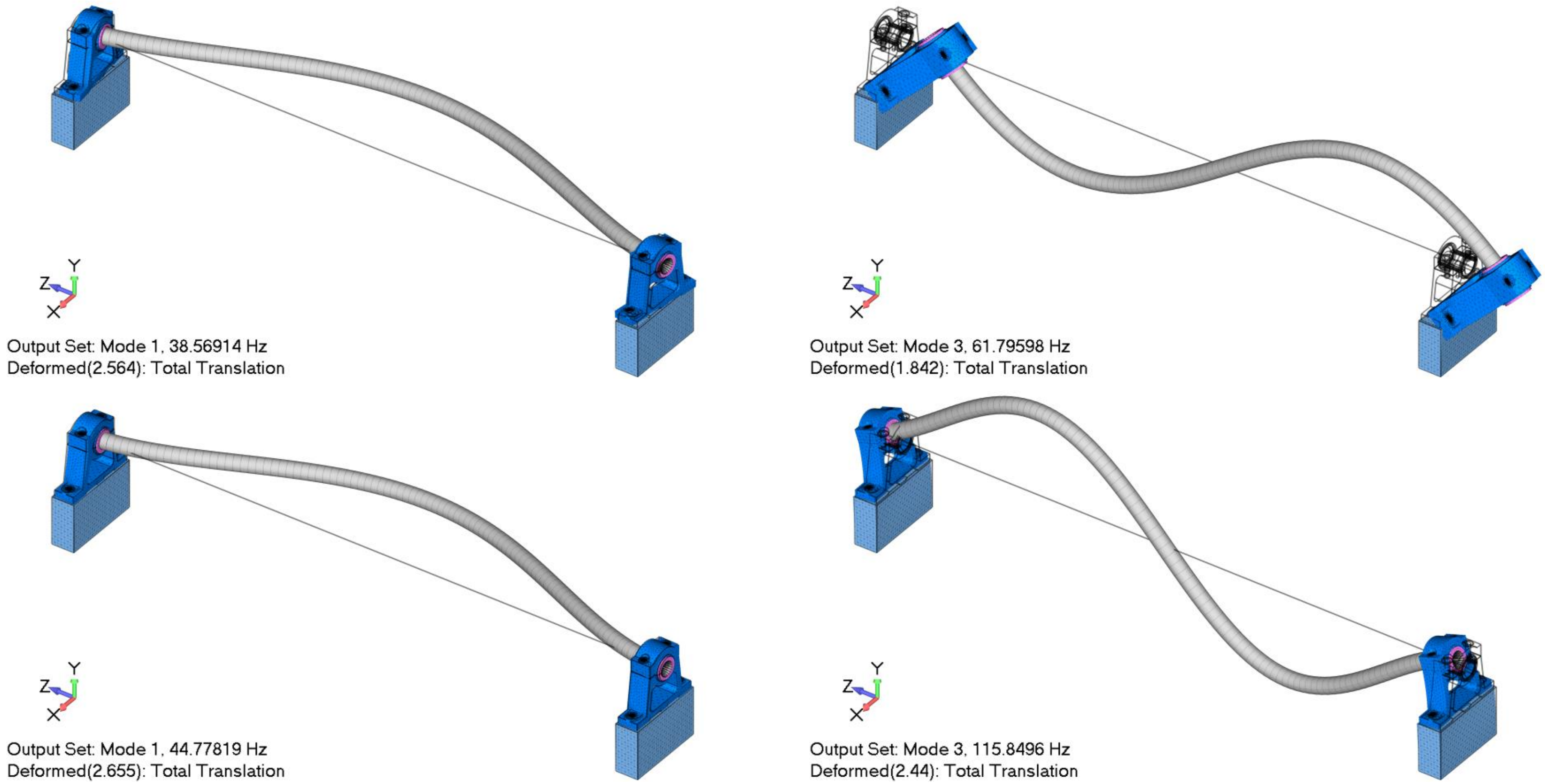


Figure 11: Improved natural frequencies with a bolt preload sub-case

3.1.2 Multi-Step Analysis (Residual Stresses)

The multi-step analysis approach provides a very useful capability to linear static and normal mode analysis. This same approach can be applied to the static nonlinear solver. Whether you want to pre-stiffen your structure (with a press-fit, shrink-fit, thermal load, etc.) or investigate residual stresses and deflections (after the structure has been plastically deformed), the multi-step analysis is the tool you want to use.

Just like the Eigenvalue analysis, incorporating additional distinct loading steps requires the addition of sub-cases. Each sub-case can have its own set of boundary conditions. In this example, we are going to pick up where the plasticity example left off (Figure 12).

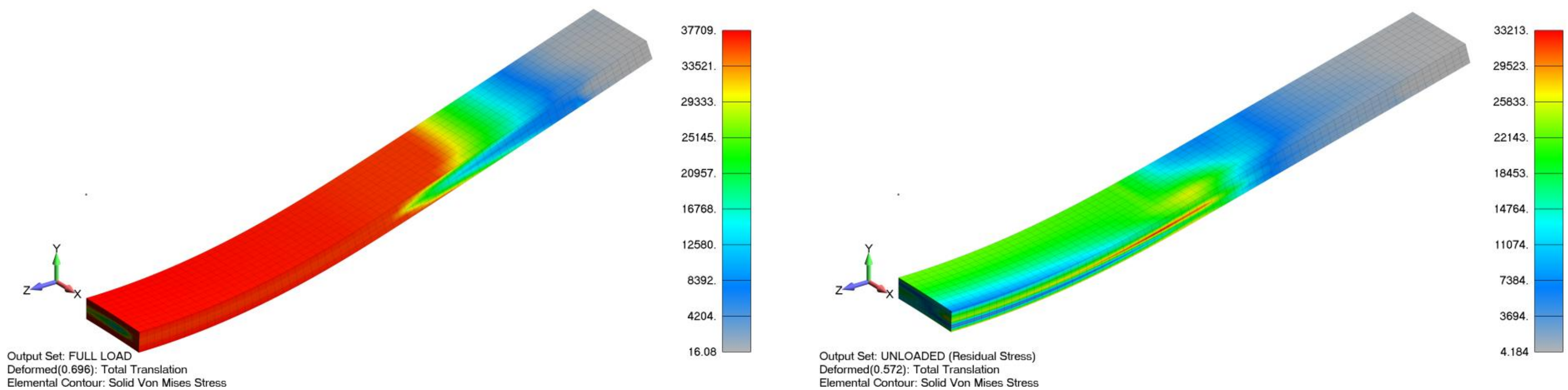


Figure 12: The four-point bending test model at maximum load and with after load is removed

The steel bar from the four-point bending test has been plastically deformed. How much of the deformation will exist after the load is removed? What will the residual stresses look like?

3.1.3 Compression Only Gap Elements (Contact) with Plasticity

Linear is contact is great for its ease of application. However, as it uses the linear solver, it doesn't provide the analyst with the capability to incorporate material nonlinearity. If you need to model both contact and plasticity, as demonstrated in Figure 13, you're going to need to spend a bit more time meshing so you can take advantage of the gap element and the nonlinear solver.

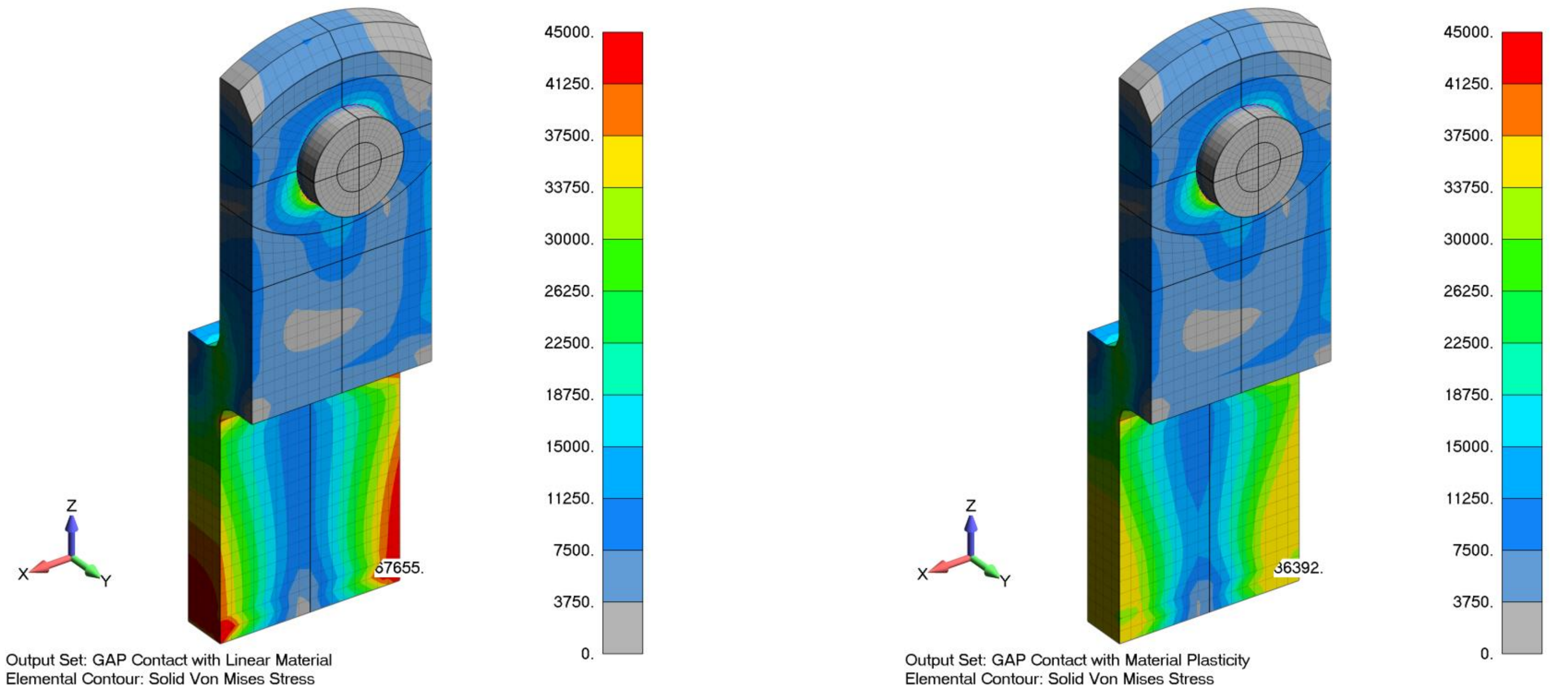


Figure 13: Pin clevis contact with material plasticity

Be aware of the limitations of gap element:

"There is no geometric nonlinear behavior, which implies that the orientation of the contact plane does not change during deflection. (The physical shape of the two contact surface would have to be specified and it would require the solution of a difficult analytic problem to determine the location of the actual contact point or points.)

3.1.4 TENSION ONLY GAP ELEMENTS (CABLES)

While gap elements are most commonly used to simulate contact, they can also be used for other handy idealizations. Since different compression and tension stiffness values can be used for the gap elements, one could model structural elements that only carry tensile loads (think straps, cables and chains). In this example, we will stabilize a radio tower with guy wires using gap elements.

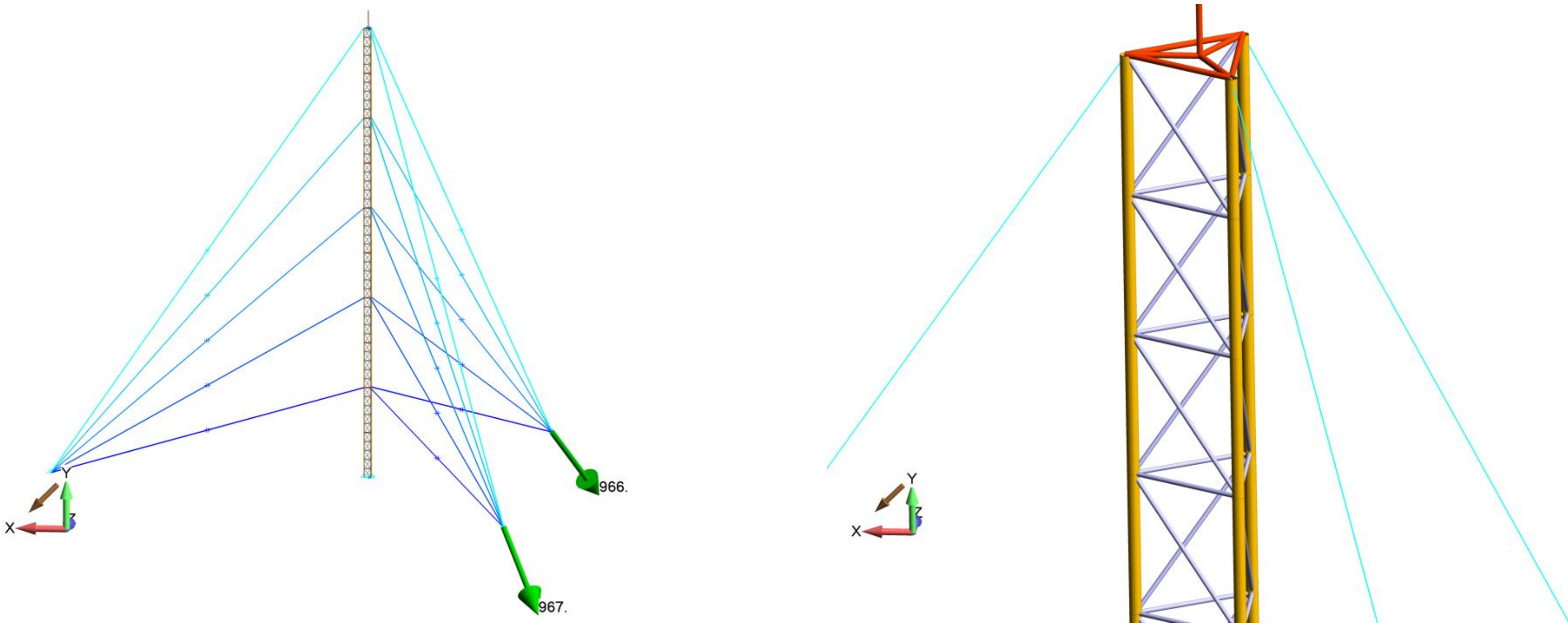


Figure 14: The tall narrow radio tower is stabilized with guy wires

4. APPENDIX

4.1 The "DOs" and "DON'Ts" of Nonlinear Analysis

(From the Basic Nonlinear Analysis User's Guide, Nonlinear Characteristics and General Recommendations)

- The analyst should have some insight into the behavior of the structure to be modeled; otherwise, a simple model should be the starting point.
- Substructuring should be considered for the modularity of the model and/or synergism between projects and agencies involved.
- The size of the model should be determined based on the purpose of the analysis, the trade-offs between accuracy and efficiency, and the scheduled deadline.
- Prior contemplation of the geometric modeling will increase efficiency in the long run. Factors to be considered include selection of coordinate systems, symmetric considerations for simplification, and systematic numbering of nodal points and elements for easy classification of locality.
- Discretization should be based on the anticipated stress gradient, i.e., a finer mesh in the area of stress concentrations.
- Element types and the mesh size should be carefully chosen. For example, avoid highly distorted and/or stretched elements (with high aspect ratio); use CTRIA3 and CTETRA only for geometric or topological reasons.
- The element net distortions have to remain small.
- Rigid body elements (RBEi, RBAR, RROD entries, etc.) do not rotate in geometric nonlinear analysis.
- Multipoint constraints (MPCs) remain linear, the user-defined constraint equations don't change automatically in geometric nonlinear analysis. In nonlinear static analysis, you may change the MPCs from subcase to subcase. Then, the changes in the MPCs are accounted for incrementally.
- Offsets in the CBEAM, CTRIA3 and CQUAD4 elements are not allowed in combination with nonlinear material.
- The results in linear and nonlinear buckling with offsets may be incorrect.

4.2 Background

4.2.1 F = K*U

The following section describes how the finite element method works. It shows how basic mechanics are used to generate the finite element displacements and stresses from a structure.

FEA is based on the displacement method, which boils down to:

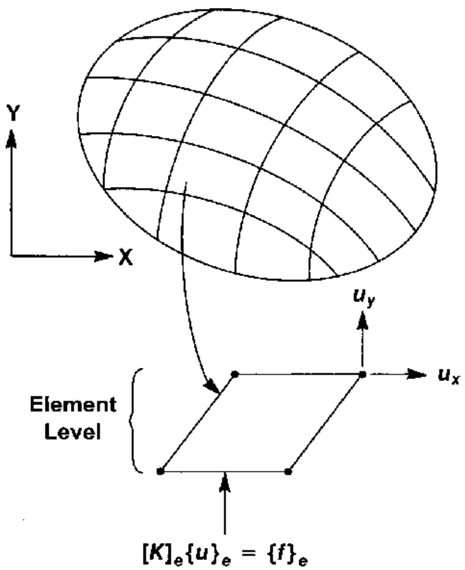
$$\{F\} = [K]\{u\}$$

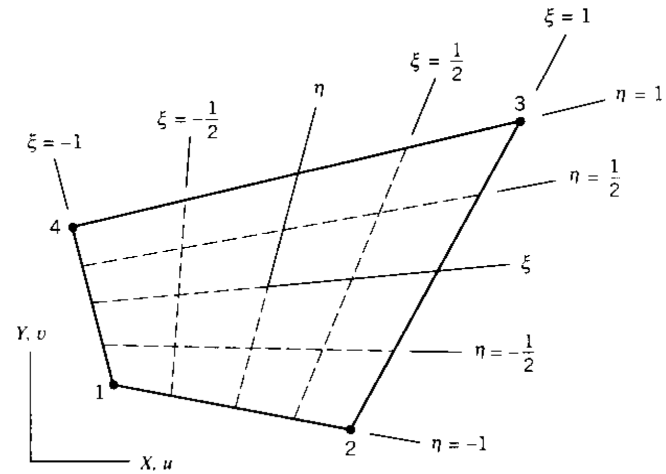
Typically, one knows something about the loads (F) that are applied to the structure and likewise its stiffness (K). The unknowns are the displacements (u) within the structure after a load is applied. Hence, the method inverts the stiffness matrix and solves for the displacements. With the displacements, one can calculate strains and with the strains you have stresses and so on and so forth.

4.2.2 Isoparametric Elements

Isoparametric (having the same parameters under different coordinate systems) are the bedrock of modern FEA. Simple functions are used to discretize oddly shaped surfaces or volumes. The basis of this method is given in the subsequent slides. Although the theory is given in 2-D it can be directly leveraged into the third dimension.

One starts with a random region that is normalized into a -1 to +1 coordinate system and two formulas that use a simple linear shape function to define interior coordinates and interior displacements:





$$x_{xp} = \sum_{i=1}^{4} N_i(\xi,\eta)x_{xi}$$

$$u_{xp} = \sum_{i=1}^{4} N_i(\xi,\eta)u_{xi}$$

N_i is known as the shape function, which does double duty as the interpolation function for both coordinates (x) and displacements (u). This is the "iso" in the isoparametric. With these formulas we can map displacements in the interior of our element and also map any coordinates. An example of a linear shape function for a four-node quadrilateral element :

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta)$$

We start with basic mechanics and apply the isoparametric method to these equations.

Step 1: Satisfy static equilibrium

$$\sum F = 0$$

Step 2: Relate strain to displacements (simple 2D example)

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \partial/\partial_x & 0 \\ 0 & \partial/\partial_y \\ \partial/\partial_y & \partial/\partial_x \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}; \qquad \varepsilon = \partial u$$

Step 3: Incorporate the shape function

This is where it gets a little complicated. To get our generalized displacements (u, v), the shape functions discussed on the prior slide are used to take corner point displacements (nodes) u_i and v_i and generate displacements anywhere within the element.

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \cdots \\ 0 & N_1 & 0 & N_2 & \cdots \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ \vdots \end{Bmatrix}; \qquad u = Nd$$

Step 4: Relate strain to displacements (using the B matrix)

Matrix "B" is called the strain-displacement matrix and is common FEA matrix jargon. The concept is that you are using the shape function to determine the "strain characteristics" within the quadrilateral element.

$$\varepsilon = \partial N d; \qquad \varepsilon = Bd; \qquad B = \partial N$$

Step 5: Relate stress to strain

$$\sigma = E\varepsilon; \qquad \sigma = EBd$$

Step 6: Relate force to stress

$$F = E\varepsilon A; \qquad F = EBdA$$

Step 7: Relate force to displacement

$$F = Kd; \qquad K = EBA; \qquad u = d$$

The pivotal part is that "EBA" is the stiffness term of the element. It is this component that is calculated to form the stiffness matrix [K].

$$\{F\} = [K]\{u\}$$

For additional reading on the subject, see the following references:

R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt, "Concepts and Applications of Finite Element Analysis," 4th Edition, 2001.

K. J. Bathe, "Finite Element Procedures," 2007.